

Energy recovery using PAT

Simão, M.¹; Ramos, H.M.¹

m.c.madeira.simao@tecnico.ulisboa.pt

¹ Civil Engineering, Architecture and Georesources Department, CERIS,
Instituto Superior Técnico, Universidade de Lisboa, Lisboa, Portugal.

PAT

Opportunity

01

Interesting and promising energy converter to improve the system energy efficiency;

02

Alternative solution to reduce the pressure in pipe systems, replacing or in conjunction with the pressure reduction valves;

03

Advantage of possible pressure control regulation and also to be green energy generator;

04

The knowledge of the operation points is difficult, when rotational speed is variable - the head and the efficiency values are difficult to predict - manufacture catalogue not have these curves;

05

This lack of knowledge as well as the difficulty to predict the best efficiency point of the pump operating in reverse mode;

Experimental tests

CFD

affinity laws through specific parameters (q , h , p , and n)

Pump working as Turbine (PAT)

Advantages

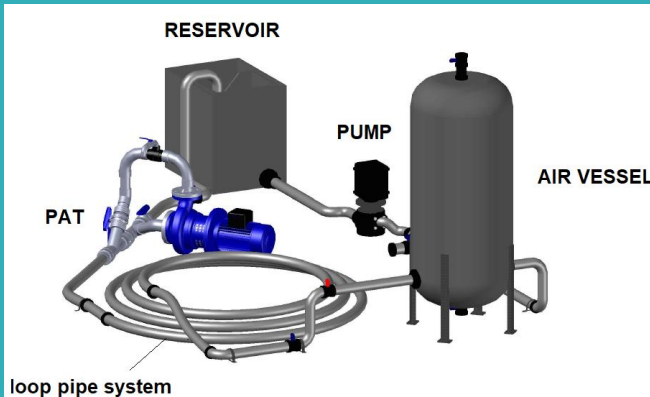
- Low investment
- Low maintenance
- Low repair costs
- Good efficiency
- Green energy
- Market availability
- Control pressure
- Flexibility



Concerns

- Energy production does not represent the principal sector of activity for operators in the water sector and in industry;
- Micro hydro power design and PAT selection are reserved to few specialists;
- The potentiality of energy recovery in pressurized networks is far to be fully conceived;
- New technologies for energy recovery in pressurized networks are often unrealistic or bad solutions;

Experimental set-up



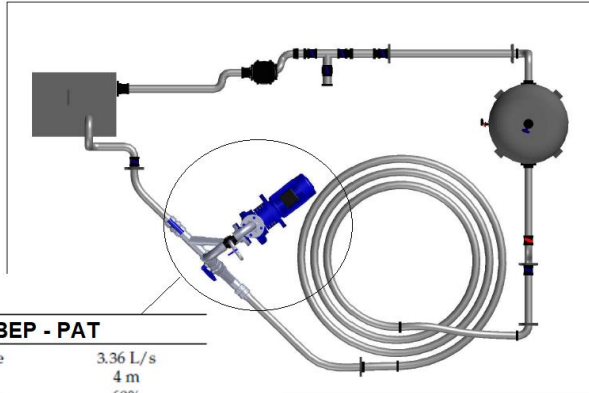
- ✓ A recirculating pump since this is a loop pipe system;
- ✓ An air-vessel tank. Allows to regulate the flow and pressure in order to reach the steady flow conditions;
- ✓ Free surface tank that keeps the recirculating flow constant in the pipe system.
- ✓ An electromagnetic flowmeter located at upstream;
- ✓ 2 pressure transducers used to register the pressure variations in a picoscope acquisition data system.



Experimental tests

PAT characteristics

TOP VIEW



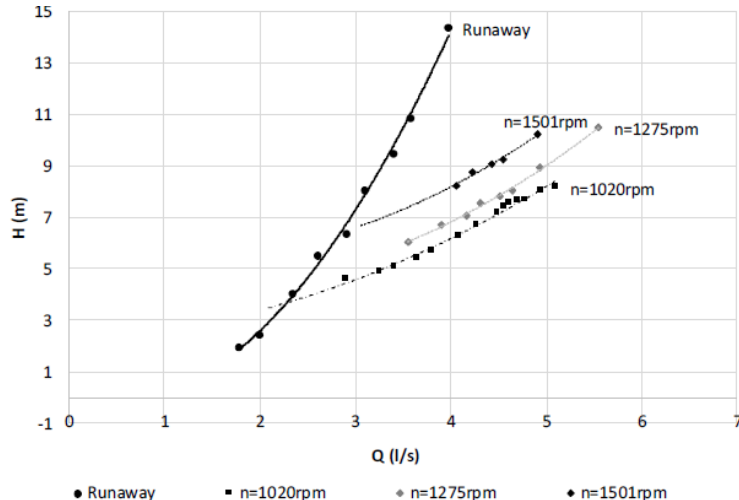
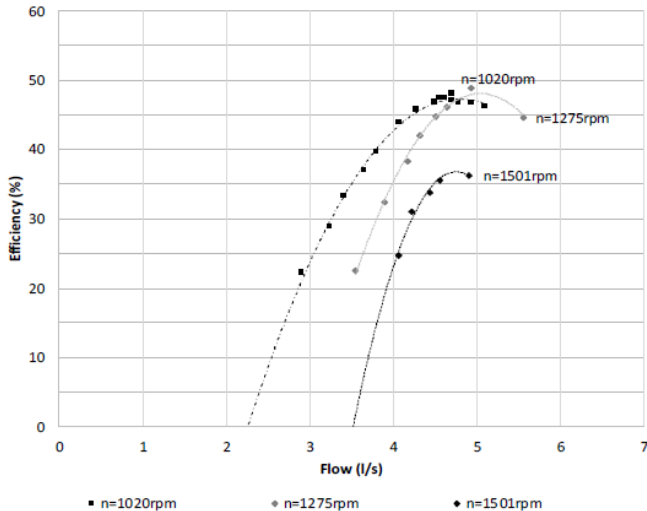
BEP - PAT

Discharge	3.36 L/s
Head	4 m
Efficiency	60%
Power	0.08 kW
Rotational speed	1020 rpm
Specific speed	51 rpm (m, kW)

- The tested and simulated machine is a centrifugal pump working as turbine with a radial impeller;
 - When the machine is operating in the turbine mode, the best efficiency point (BEP) is 3.36 l/s and 4 m w.c., reaching an efficiency of 0.60;
 - For these values, the specific speed of the machine is 51 rpm (m, kW) and its nominal rotational speed is 1020 rpm.
- The PAT model is from KSB.

Experimental tests

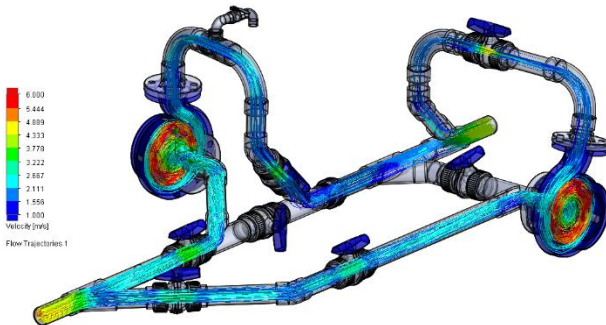
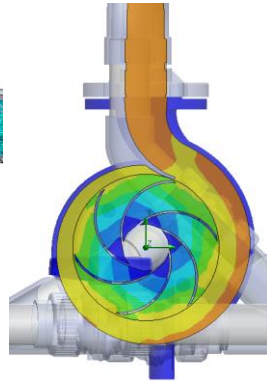
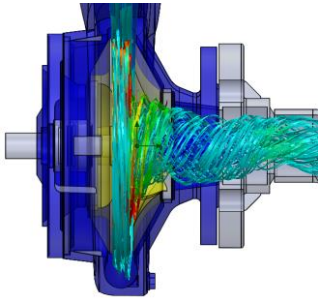
PAT characteristics



- The experiments were carried out for different rotational speeds and different flows in steady state conditions for different rotational speeds: 1020, 1275 and 1500 rpm;
- For each rotational speed, different operation points were recorded with flow values between 2.9 and 5.0 l/s;
- The below flow limit was established by the runaway curve, where the flow decreases when the speed increases for this type of impeller.

Numerical Models

Importance of CFD



01

Simulation of a complete set of boundary conditions

Experimental tests are incomplete or too expensive

02

Optimization

To increase efficiency

03

Know-how

Influence of surge conditions; hazards situations; leaks; cavitation

04

Plainness/clarification

Examination of the flow field and flow pattern in areas where it is not possible to visualize/understand

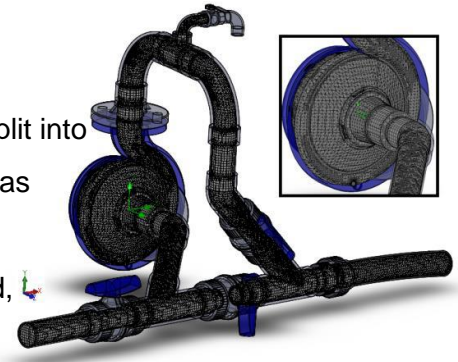
Numerical Models

CFD software works with CAD integration, which means that it takes geometrical data from a CAD program directly, suitable for parametric analysis.

The **mesh** was determined by the computational domain using the following procedure:

- 1) the computational domain was divided into slices;
- 2) the mesh cells intersecting with the solid/fluid interface were split uniformly into smaller cells in order to capture the solid/fluid interface with the specified mesh cells;
- 3) each of the basic mesh cells intersecting with the solid/fluid interface were split uniformly into cells;
- 4) each of the cells intersecting with the interface was in turn split into 8 cells of next level, and so on, until the specified cell size was attained.

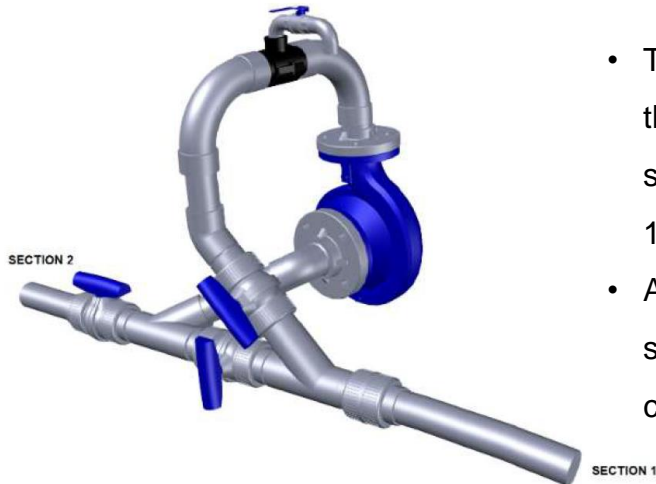
The rectangular **computational domain** was automatically constructed, enclosing the solid body with the orthogonal boundary planes.



CFD Model

Boundary conditions

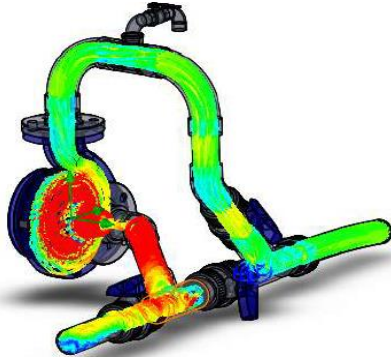
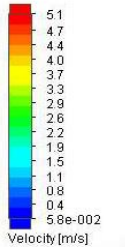
- In the CFD simulations, the static pressure (section 2), the volume flow rate (section 1) and the impeller rotational speed (section 3) boundary conditions are assigned to the inflow and outflow boundaries in the geometric model;
- The first initial boundary condition is set upstream of the pipe with inlet volume flow varying from 2.90 l/s to 5.00 l/s;



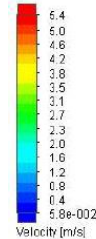
- The second initial boundary condition is set at the inside surface for the outlet pipe with a static pressure varying between 112000 and 113500 Pa;
- As for the rotational speed, 3 rotational speeds of 1020, 1275 and 1500 rpm are considered.

CFD Results

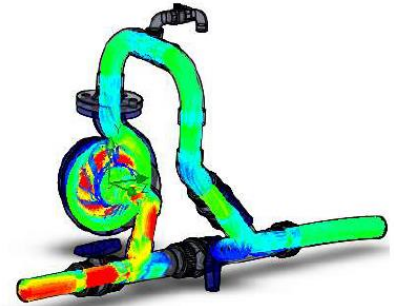
The hydrodynamic of the flow is characterized by the presence of curves and elbows at upstream of the PAT.



Flow Trajectories 1

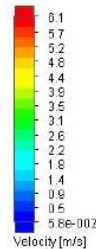


Flow Trajectories 1

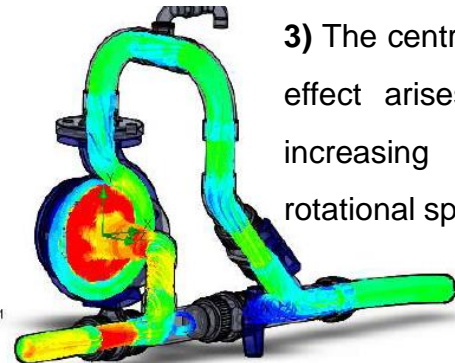


1) Reduction in the velocity magnitude, upstream and along the impeller, caused by the energy promoted by the flow to the impeller;

2) Vortex that forms at the exit of the rotor, and extends downstream thereof is associated with turbulence, and hydrodynamic instability, which effects result in pressure fluctuations and efficiency losses;

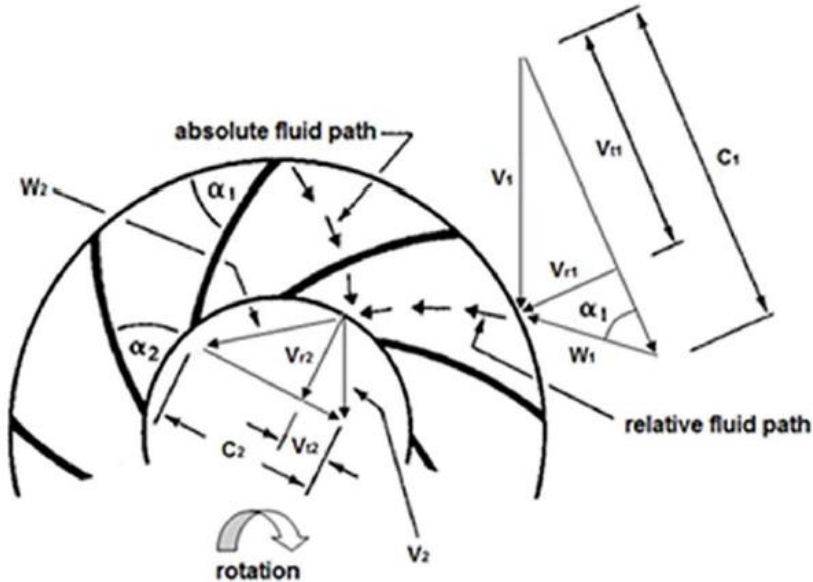


FlowTrajectories 1

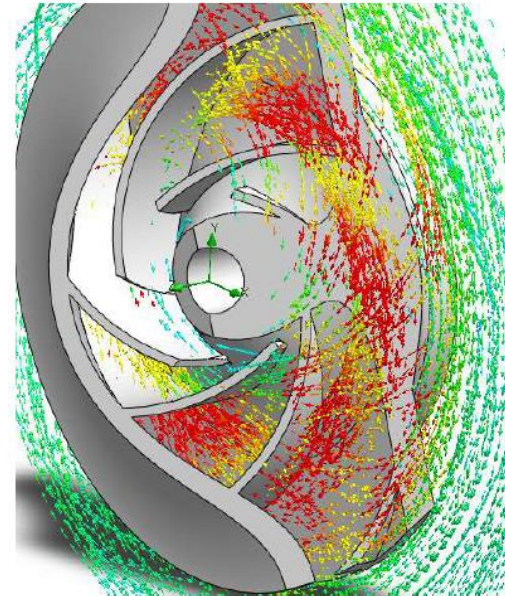


3) The centrifugal force effect arises with the increasing of the rotational speed.

CFD Results

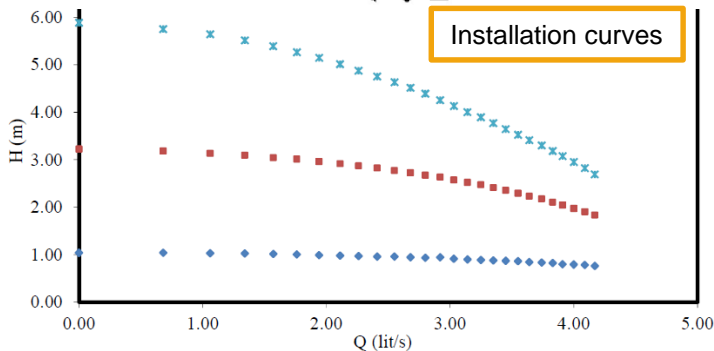
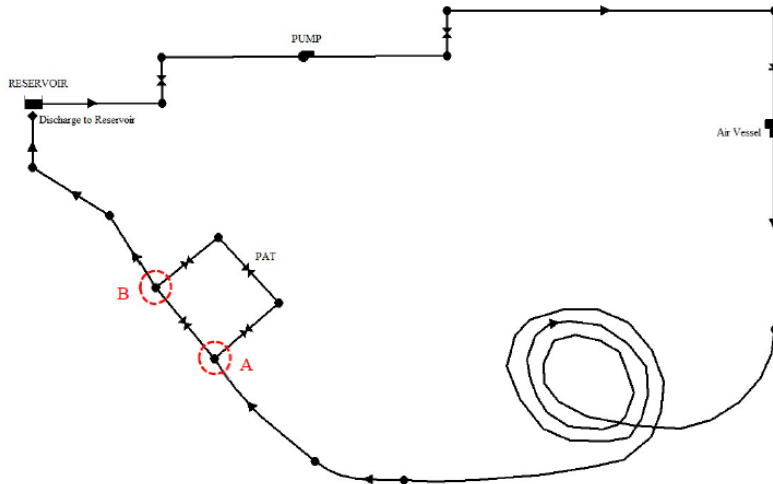


Velocities fields of the flow inside the impeller at each particle position.



Distribution of the velocity vectors as the flow enters and leaves the impeller: The velocity vectors can be decomposed into absolute velocity (V), relative velocity (W) and peripheral velocity.

EPANET model

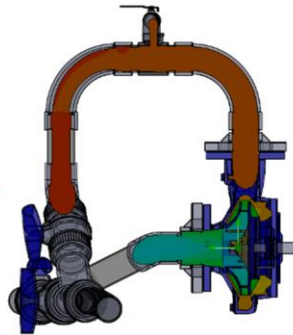
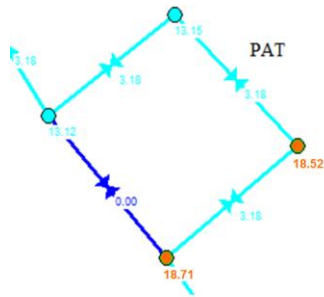
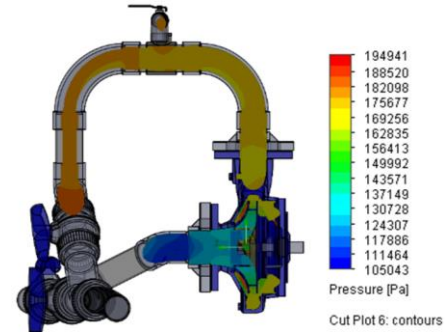
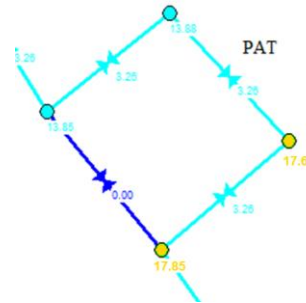


The EPANET model was prepared based on the experimental model considering the pipe lengths, diameters, minor head losses and pipe:

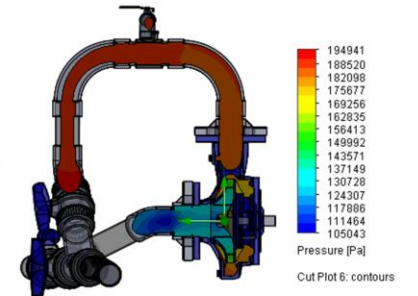
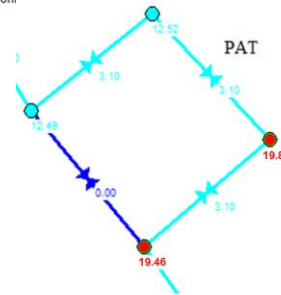
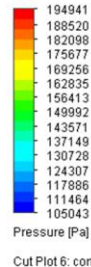
- The pump curve was included in EPANET for the pump operation;
- A general purpose valve (GPV) was selected using the turbine curve as the operational curve of the valve;
- The initial head of the air vessel (H_0) was changed from 10 to 40 m, in order to get the different operating conditions in the PAT i.e., (N and H) which depends on the head difference between points A and B.

Comparisons

The pressure decreases from upstream to downstream as the fluid flows within the domain and along the impeller, from the inner to the outer region, as energy is transmitted to the shaft.



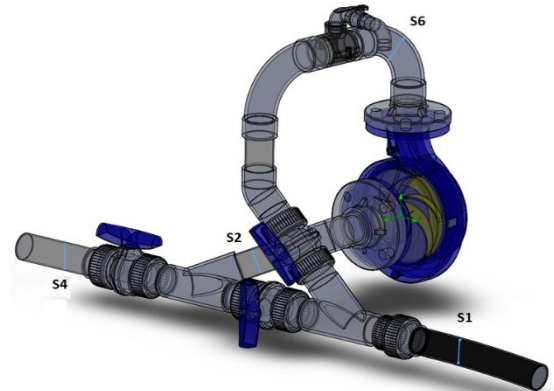
For a rotational speed of 1275 rpm and 1500, the head losses obtained by each numerical model are similar.



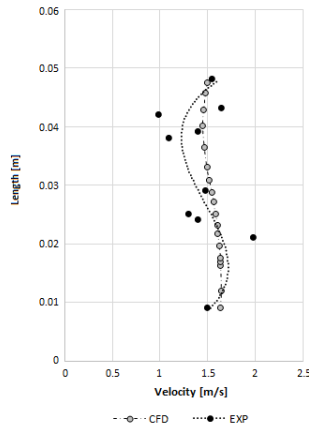
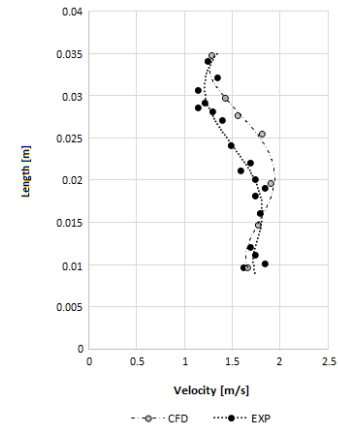
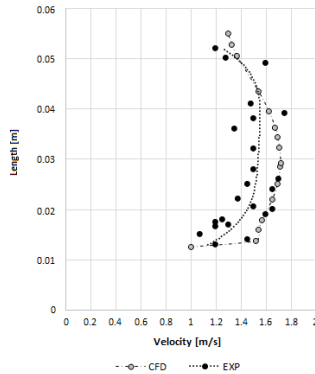
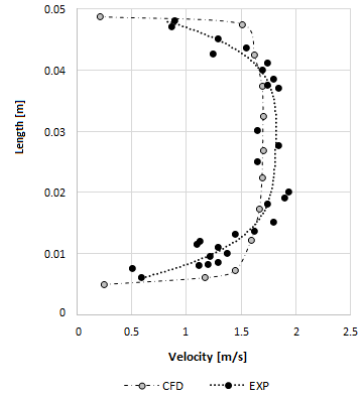
the head losses average given by EPANET from upstream to downstream of the PAT are about 4 m, against to 5.7 m from CFD model and 4.8 m from experiments.

Comparisons

velocity profiles were collected in different sections of the pipe using a **3000 serie Ultrasonic Doppler Velocimetry (UDV)**



Identify rotating flow zones, influence of bends and runner in velocity diagrams and dissipation effects



Conclusions

Understand the behaviour of the major parameters of the system under different conditions:

- ✓ EPANET results show good accordance with experiments in terms of pressure in nodes and flow rates in pipes;
- ✓ The system curve demonstrate a reduction of the PAT's net head with the increasing flow rate in all the studied cases;
- ✓ The proposed 3D CFD model gives a complete information about the velocity change in different parts of the PAT including the impeller, and the inner and outer parts of the PAT;
- ✓ The provided velocity streamlines declare the velocity reduction in the upstream vicinity of the PAT and along the impellers;
- ✓ Evident vortices were also formed in different parts of the PAT which is related to turbulence caused by pressure fluctuations;
- ✓ The typology of the machine (i.e., specific speed) influence the percentage of the head drop and the evaluation of local head losses in the simulation period;

Conclusions

- ✓ Nevertheless, this research allowed to identify the head drop in different parts of the PAT domain, for different rotational speeds as well as the velocity fields variation;
- ✓ Aside from geometry differences between CAD and the real PAT, the discrepancies of mechanical losses concerns the scale effect of frictions and bearings, the surface roughness of the tested model, the eventually entrapped air of micro bubbles in the pipe loop system and inside the PAT, responsible for additional turbulence and the swirl effects;
- ✓ The PAT's flow and head provided by EPANET and 3D CFD models are quite similar to the experiments showing the high performance of numerical models to represent the behaviour of the flow inside a PAT.

Thank you

Simão, M.¹; Ramos, H.M.¹

m.c.madeira.simao@tecnico.ulisboa.pt

¹ Civil Engineering, Architecture and Georesources Department, CERIS,
Instituto Superior Técnico, Universidade de Lisboa, Lisboa, Portugal.